

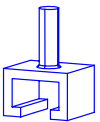
## Creating Basic Machine Parts

I-DEAS™ Tutorials: Advanced Projects

**Learn how to model these parts:**



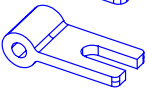
mounting plate (page 5)



drawer slide support (page 10)



tie-down strap (page 17)



support arm (page 26)



shaft bearing (page 34)

# Before you begin...

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## Prerequisite tutorials:

1. Introducing the I-DEAS™ Interface

—or—

Quick Tips to Using I-DEAS

—and—

Creating Parts

2. Sketching and Constraining

## Recommended tutorials

1. Using Sketch Planes and Understanding Sketch Pads

2. Extruding and Revolving Features

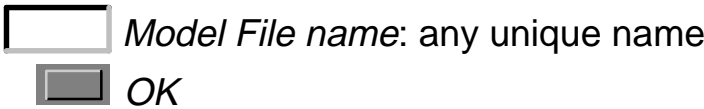
This tutorial lets you practice the skills presented in the Sketching and Constraining Fundamental Skills tutorial.

To get the full benefit of the tutorial, try building each part on your own from the dimensions shown. If you have trouble, subsequent pages give you step-by-step instructions.

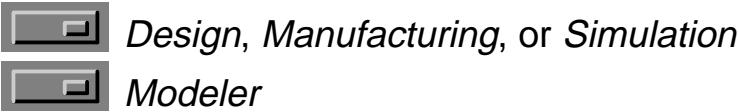
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



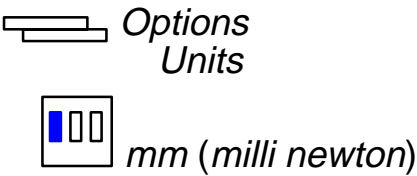
Open Model File form



Make sure you're in the following application and task:



Set your units to mm.



**Save your model file.****Warning!**

If you are prompted by I-DEAS to save your model file, respond:

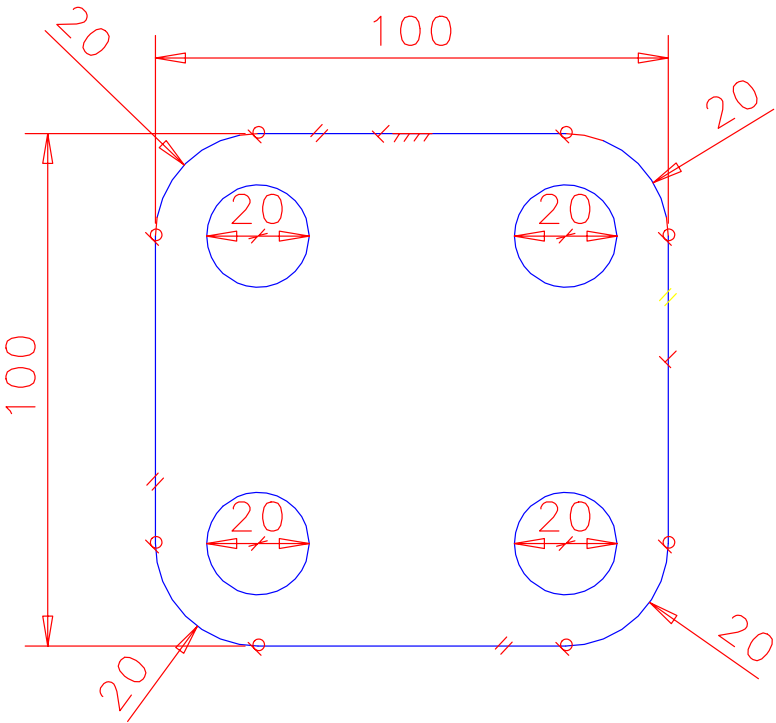
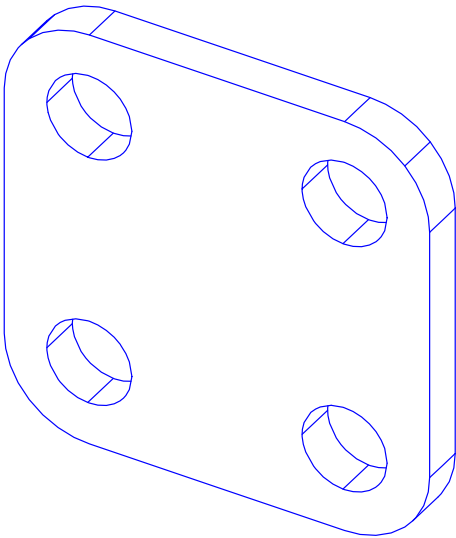


Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

**Hint**

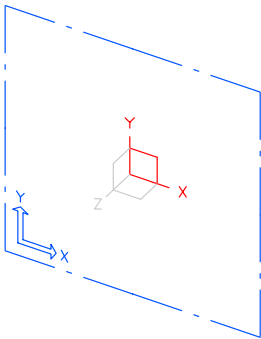
To reopen your model file to the previous save, press Control-z.



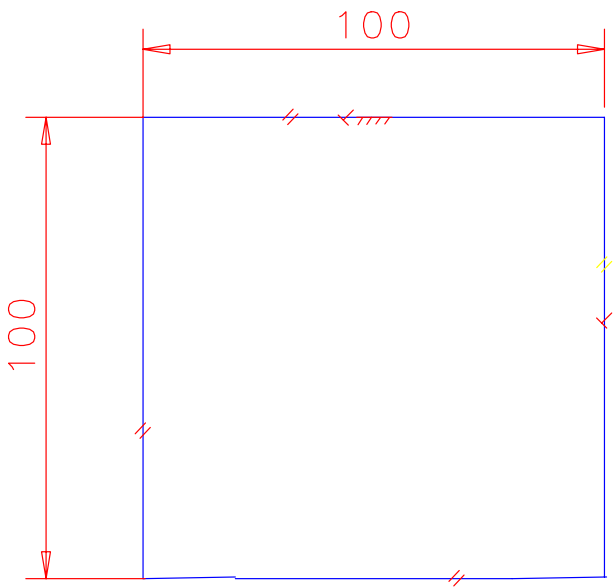
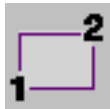
# Model a mounting plate

2 of 5

Create a part and name it “mounting plate.” Pick the XY plane to sketch on.



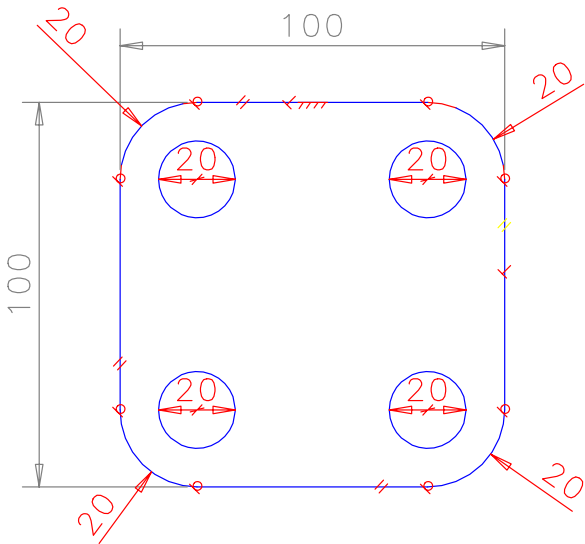
Sketch a rectangle and modify the dimensions.




## Recovery Point

 *File*  
*Save*

Fillet the corners and add the four circles.

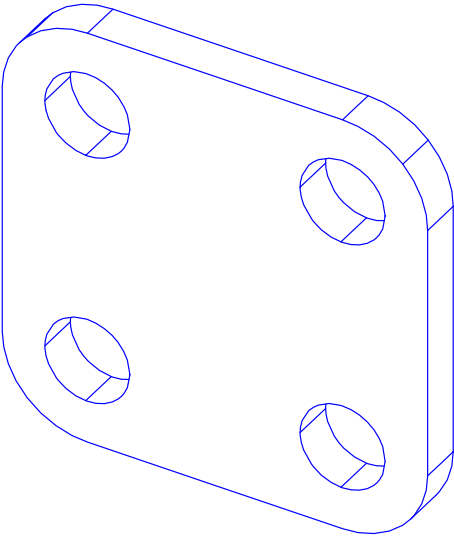
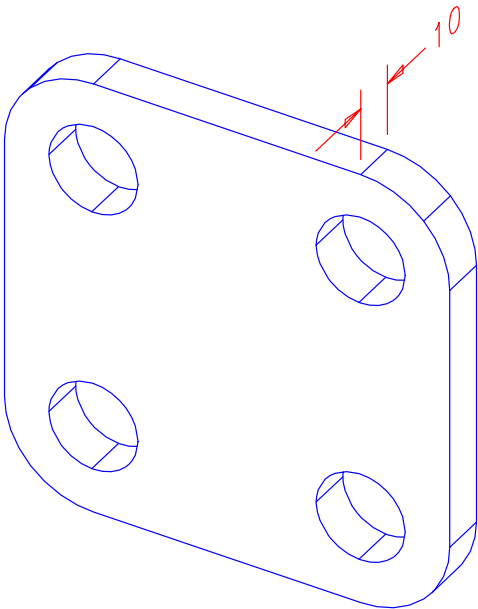


 Creating the fillets first provides a center point to locate the circle centers for the holes.

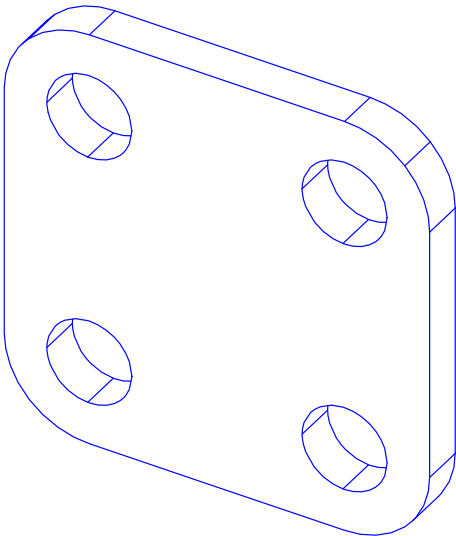
Recovery Point



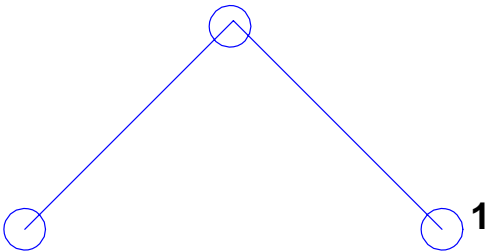
Extrude as one feature.



Finished part.

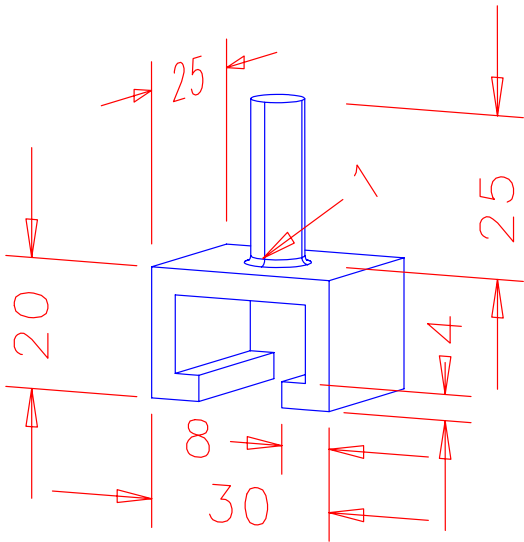
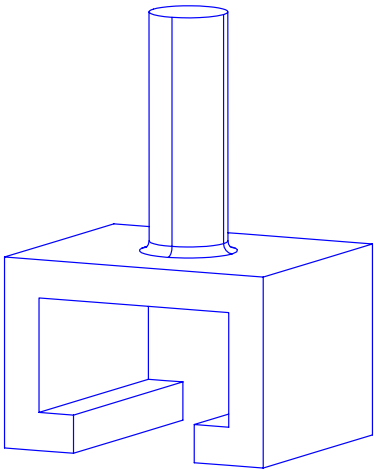


Check the history tree.



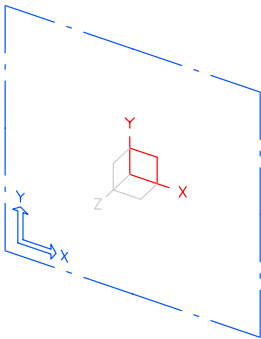
1. extrusion

You can delete the part or put it away so you have an empty workbench for the next part.

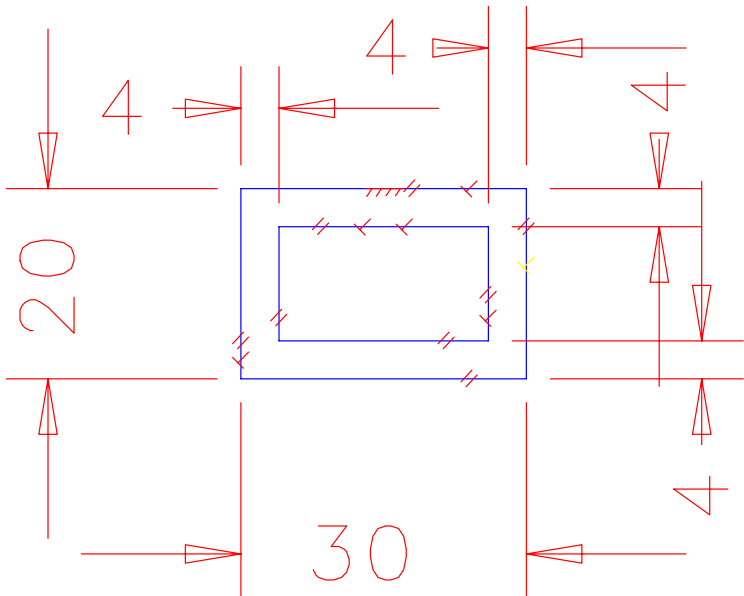
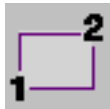


Not all dimensions are defined above. Use your own judgment to fill in missing values.

Create a part and name it “drawer slide support.” Pick the XY plane to sketch on.



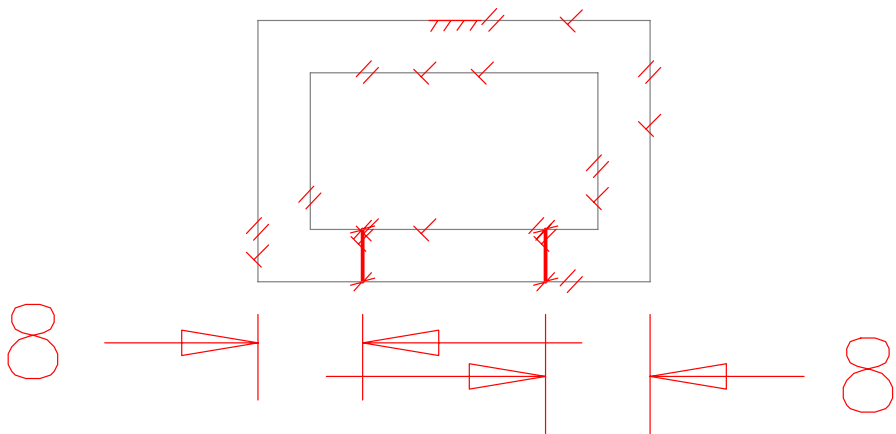
Sketch two rectangles and modify the dimensions.



Recovery Point



Add two single lines and dimension as shown.



Recovery Point

 *File*  
*Save*

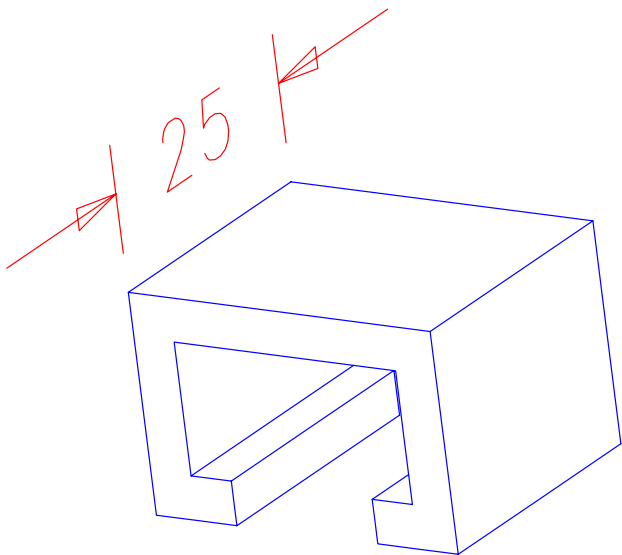
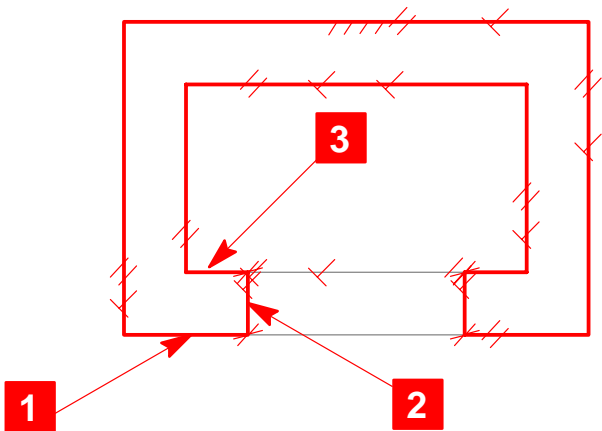
Extrude the section as shown.



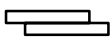
Section Options...



Stop at Intersections

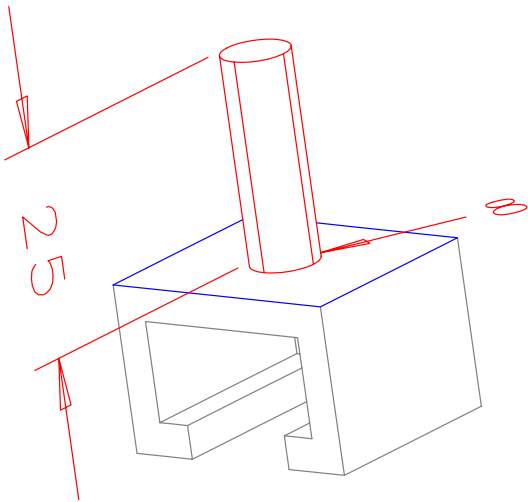


Recovery Point



File  
Save

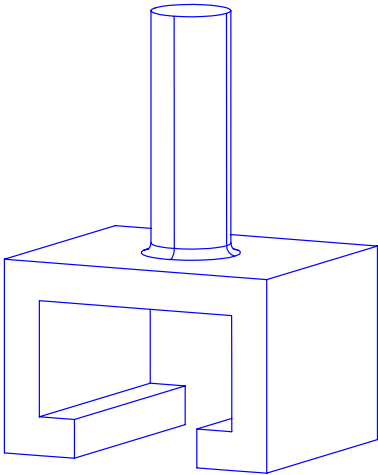
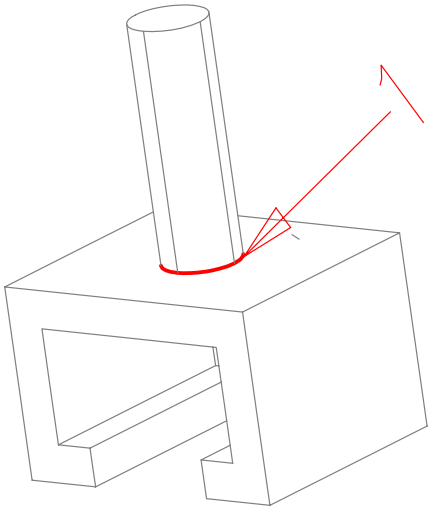
Sketch in place a circle (diameter = 8). Then extrude the circle with the value shown below.



Recovery Point



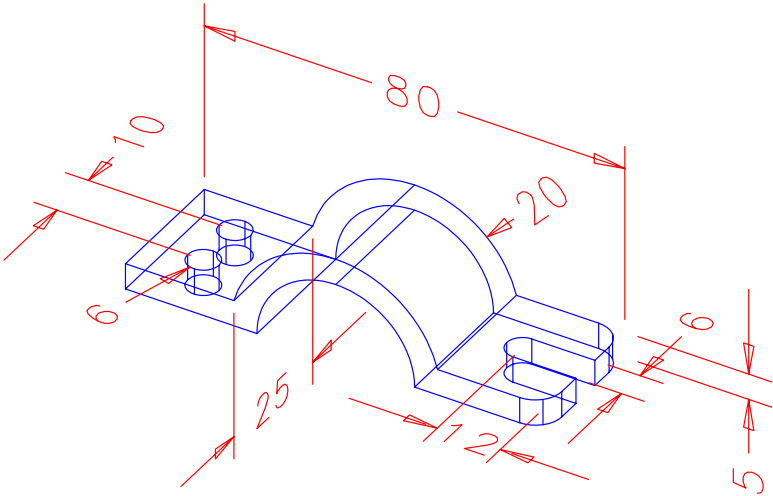
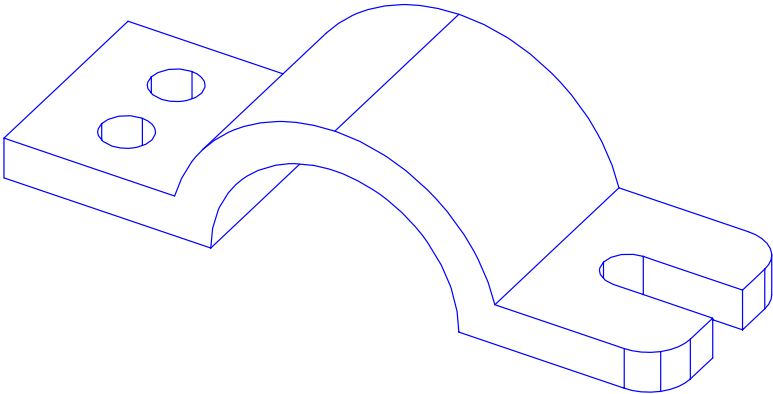
Fillet the intersection shown.



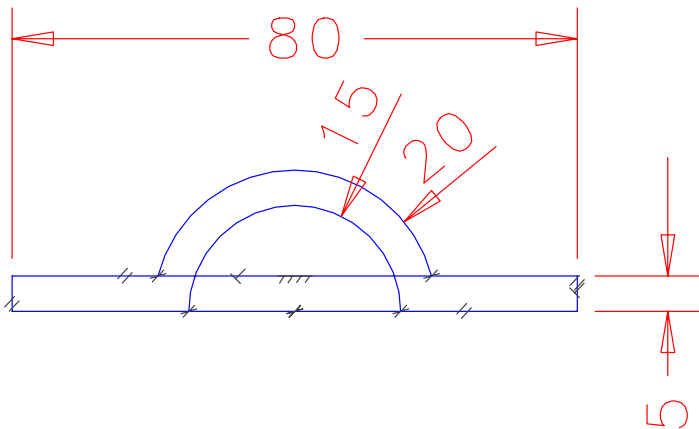
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- You can delete the part or put it away so you have an empty workbench for the next part.



Sketch a rectangle to the dimensions shown. Sketch two arcs.



The centers of both arcs are on the midpoint of the lower line. You may need to constrain the ends of the arcs to be colinear with the lines.

Recovery Point



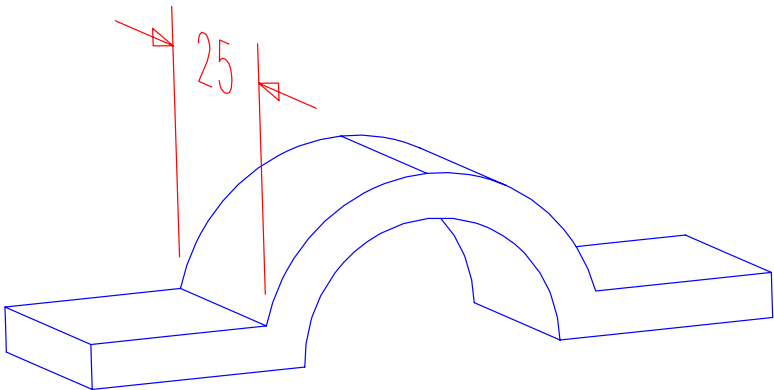
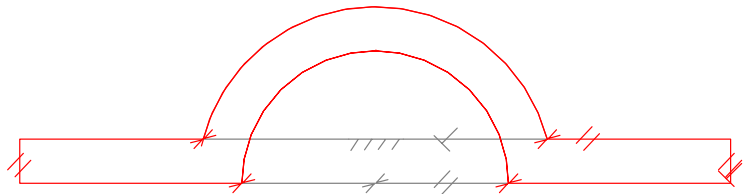
Extrude the distance shown.



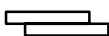
Section Options...



Stop at Intersections

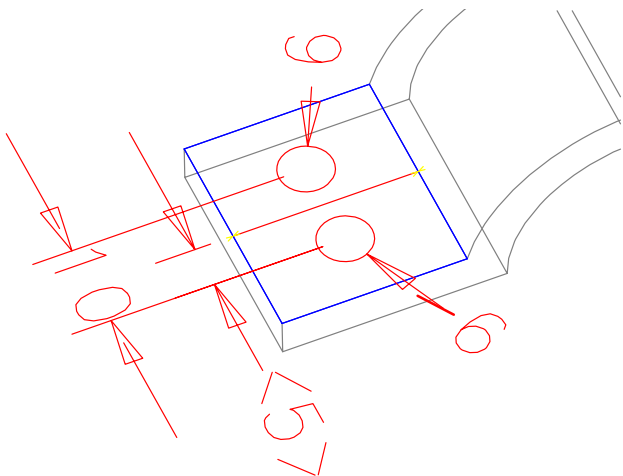



Recovery Point



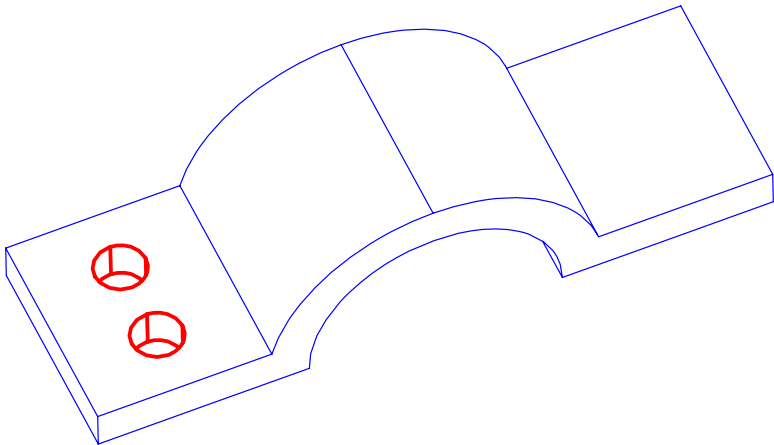
File  
Save

Sketch in place on the surface shown. Draw a center line using a single line. Draw two circles and dimension as shown.



 Match the distance to the centerline to be half the distance between the two circles.

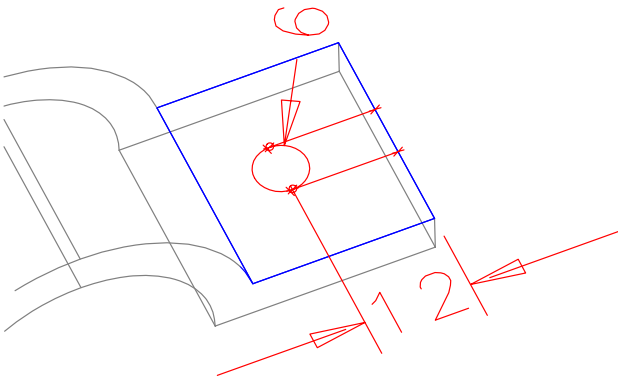
Extrude (cut) the circles through the part.



Recovery Point



Sketch in place on the surface shown. Draw a circle and two lines and dimension as shown.



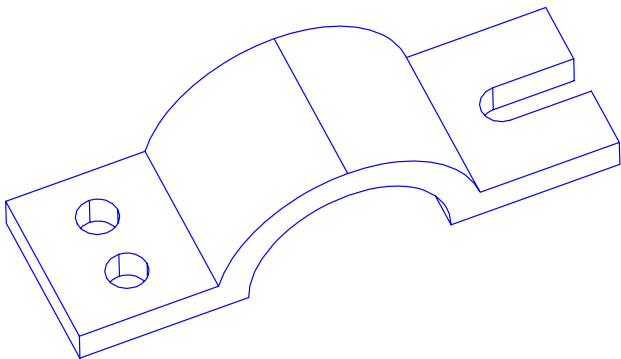
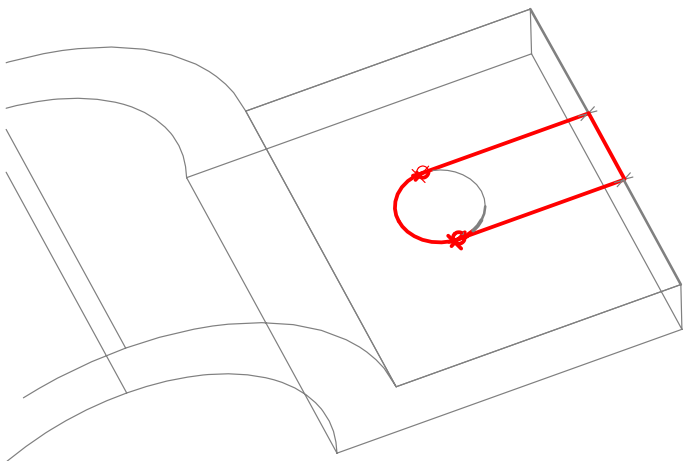
Extrude (cut) as shown.



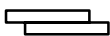
Section Options...



Stop at Intersections

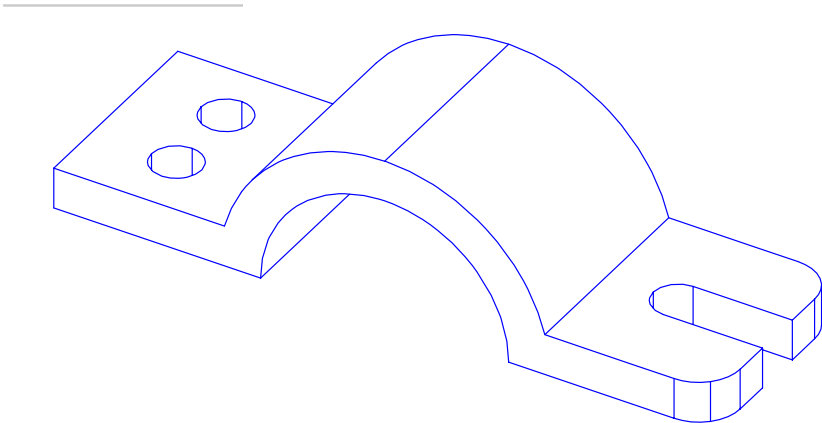
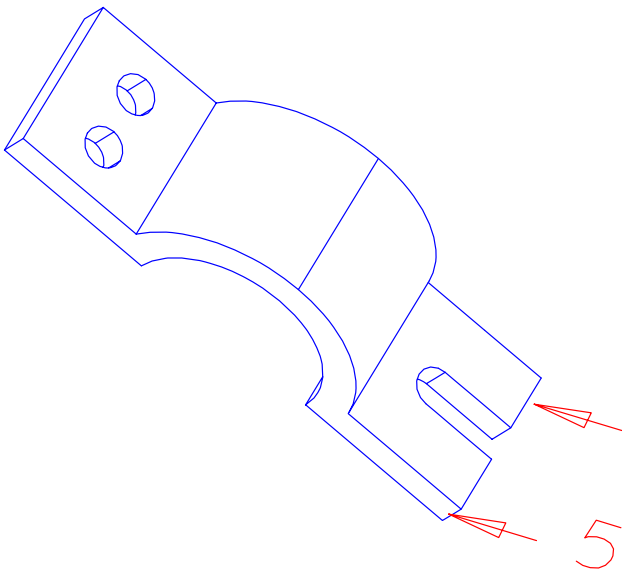


Recovery Point

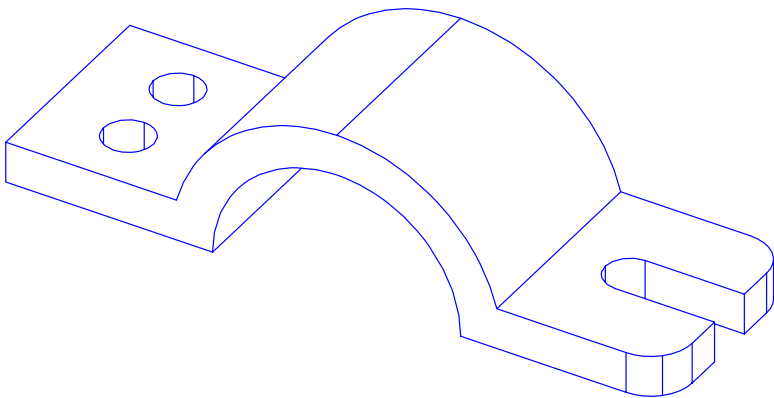


File  
Save

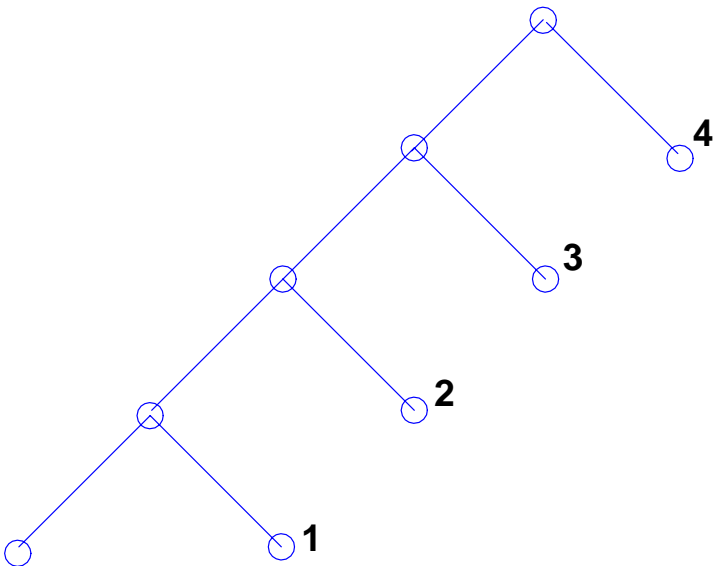
Fillet the two edges shown.



Finished part.

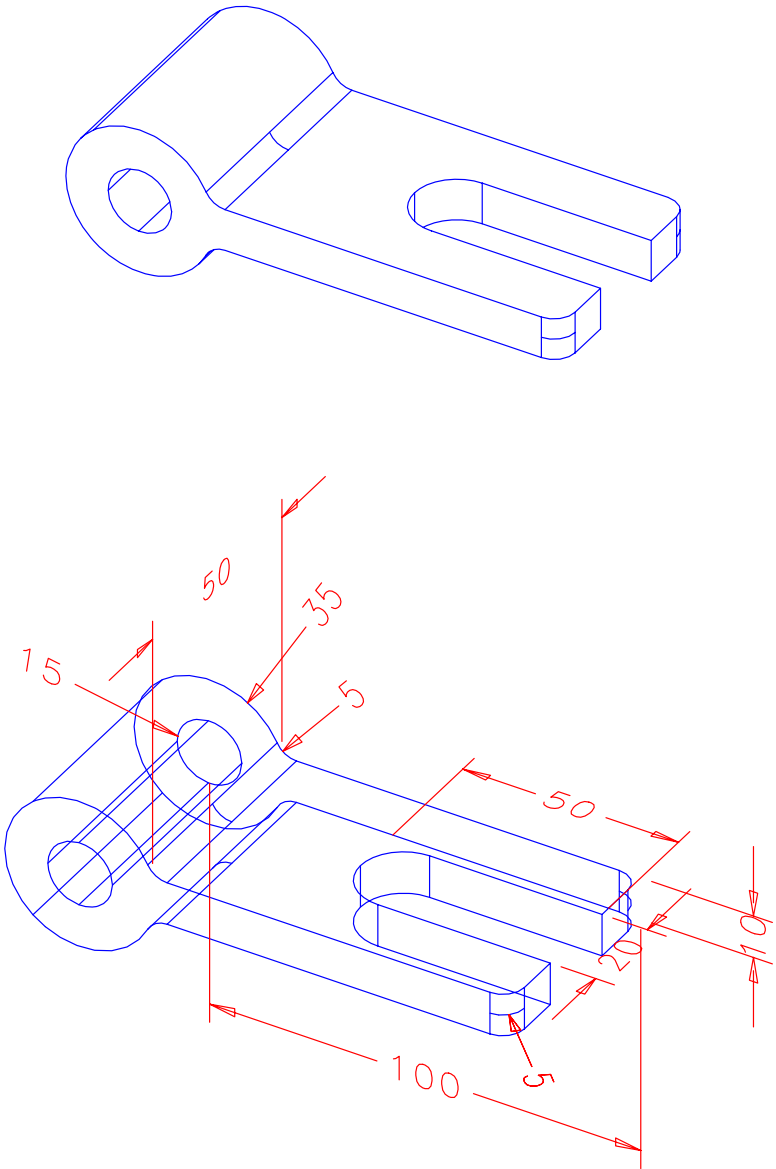


Check the history tree.



- 1. extrusion
- 2. holes
- 3. slot
- 4. fillet

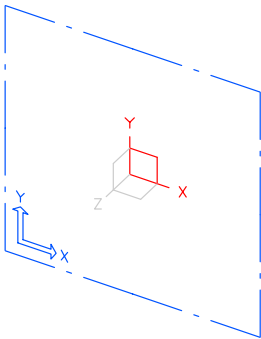
You can delete the part or put it away so you have an empty workbench for the next part.



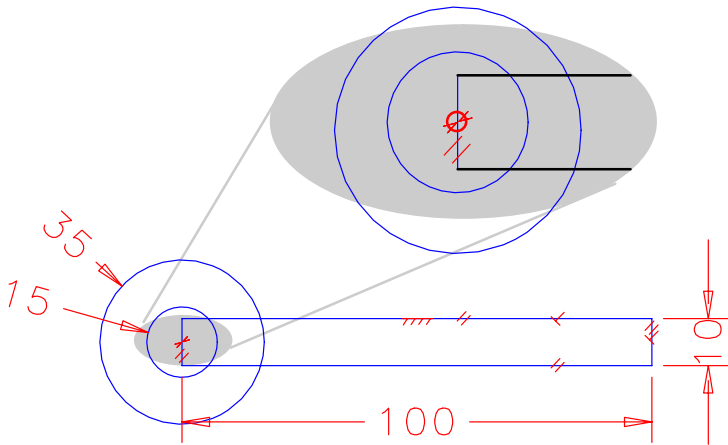
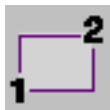
# Model a support arm

2 of 8

Create a part and name it “support arm.” Pick the XY plane to sketch on.



Create a rectangle and dimensions as shown. Draw two circles centered on the left edge midpoint.



## Recovery Point

 *File*  
*Save*

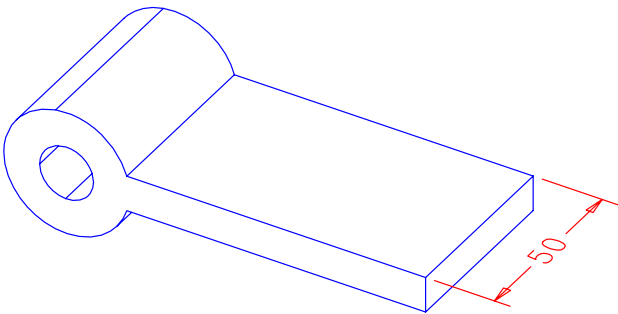
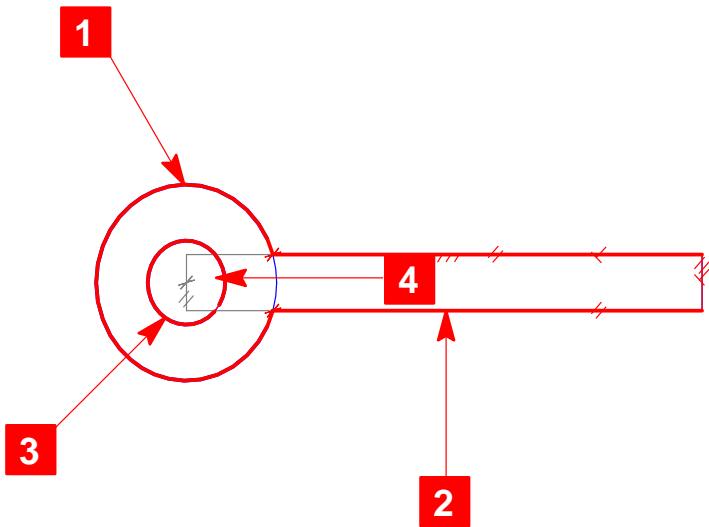
Extrude the section as shown.



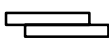
Section Options...



Stop at Intersections

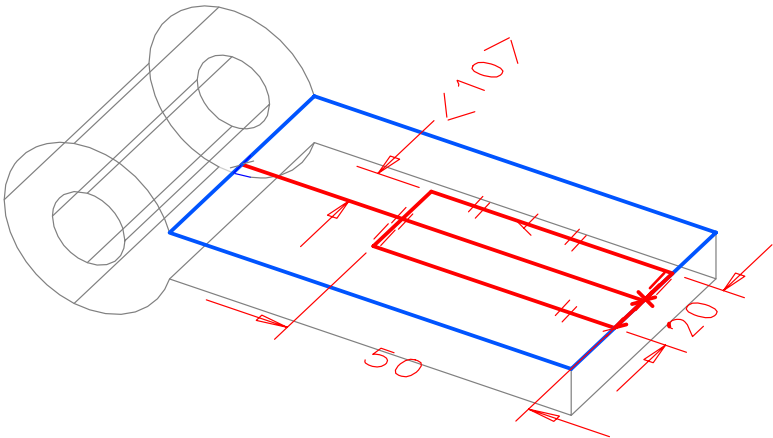


Recovery Point



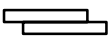
File  
Save

Sketch in place on the surface shown. Draw a center line, sketch a rectangle, and dimension as shown.



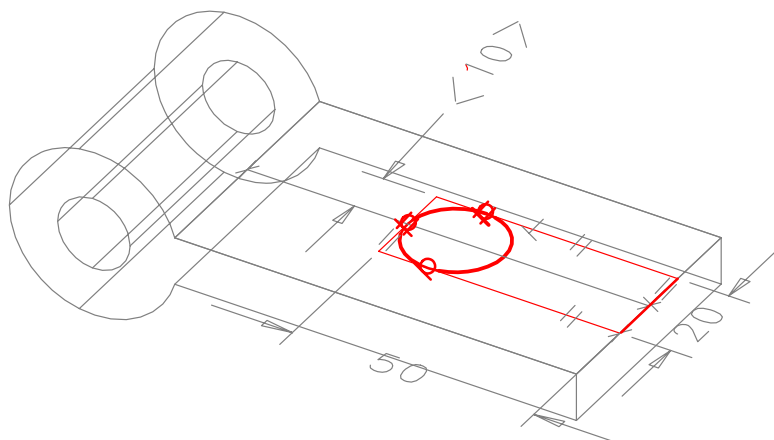
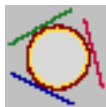
To keep the slot centered, match the distance from the center line to one side to be half the slot width.


Recovery Point



File  
Save

Draw a circle using the *Tangent to Three* icon.



 This is a good technique for creating rounded slots, since the slot length can easily be dimensioned. When you extrude (next step), do not trim the curves—use *Stop at Intersections* instead.

## Recovery Point



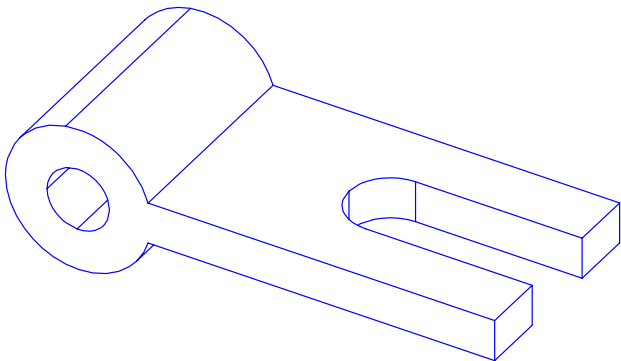
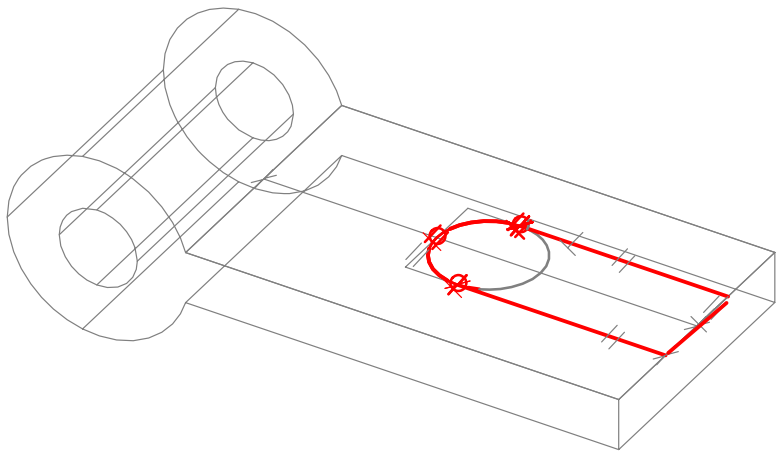
Extrude (cut) as shown.



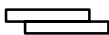
Section Options...



Stop at Intersections

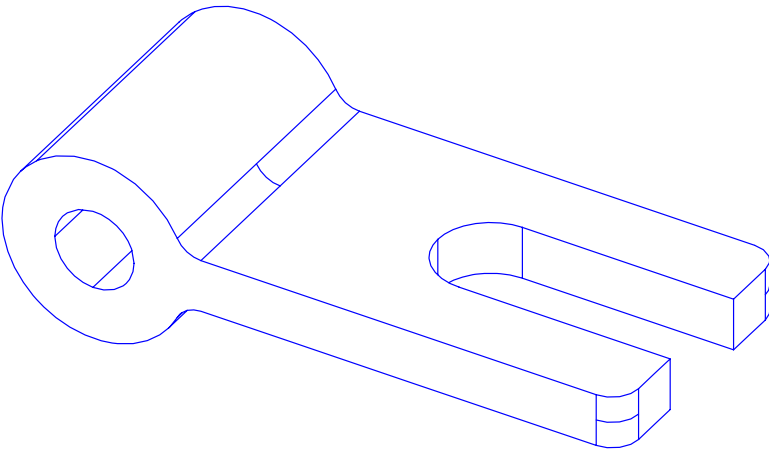
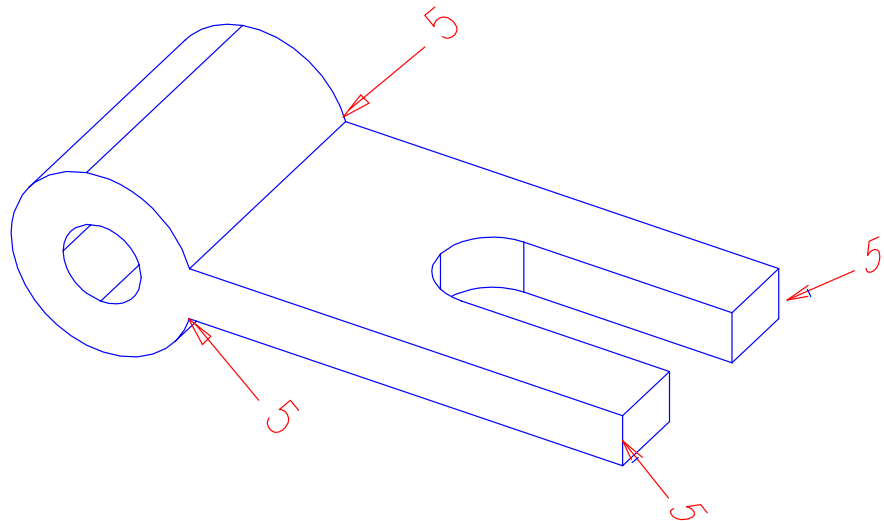


Recovery Point

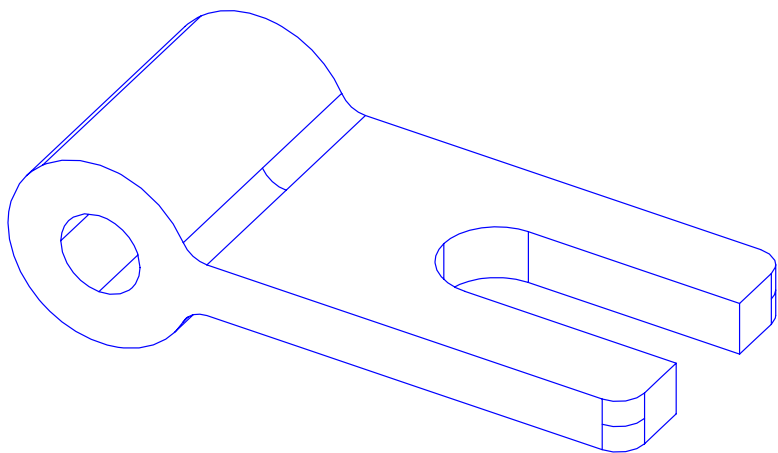


File  
Save

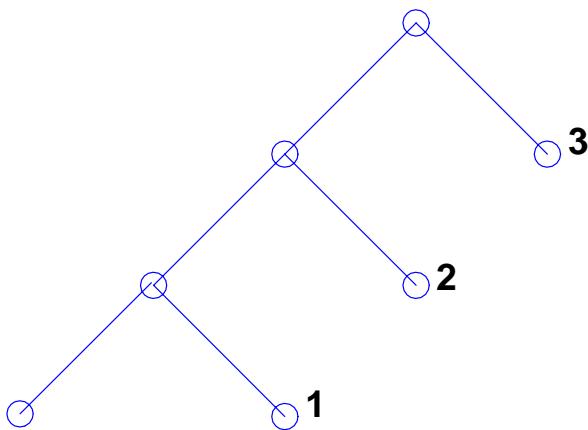
Fillet the edges as shown.



Finished part.

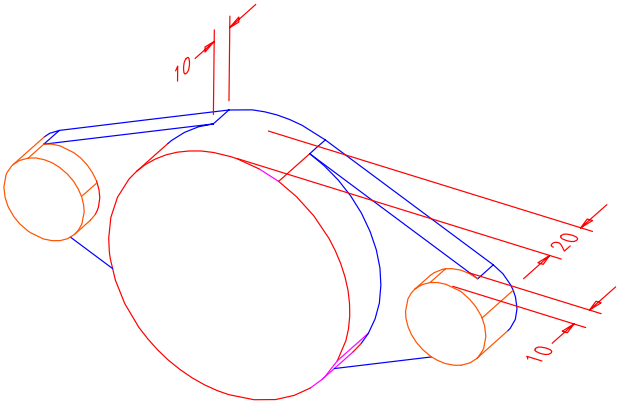
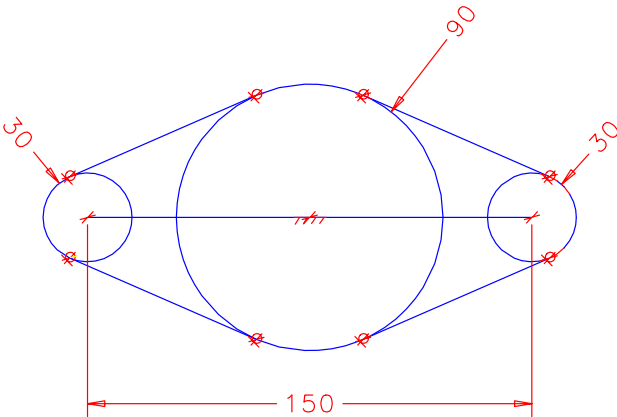
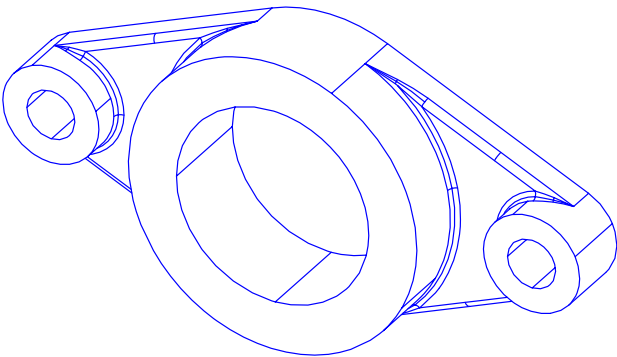


Check the history tree.

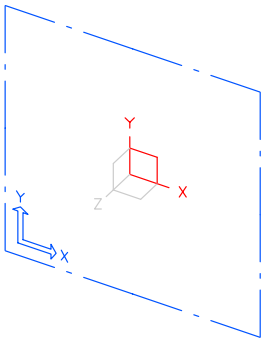


- 1. extrusion
- 2. slot
- 3. fillet

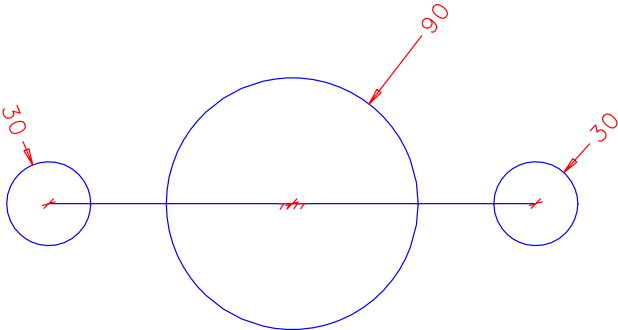
You can delete the part or put it away so you have an empty workbench for the next part.



Create a part and name it “shaft bearing.” Pick the XY plane to sketch on.



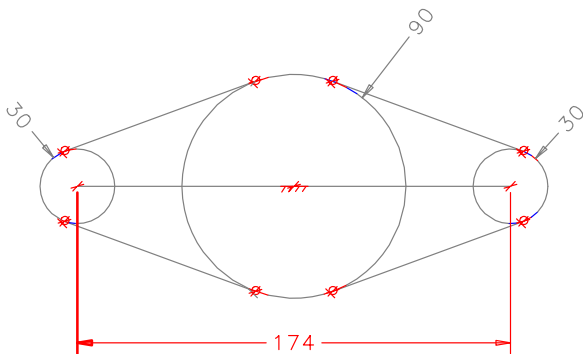
Sketch a horizontal line. Create circles located at the ends and center of the line. Modify the dimensions as shown.



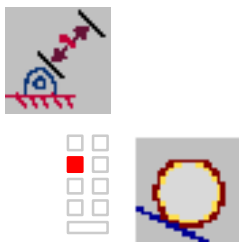
Recovery Point




Create two single lines tangent to the circles. Add dimensions.



**Hint**  
Use the *Constrain* subpanel if necessary to make sure the lines are tangent.

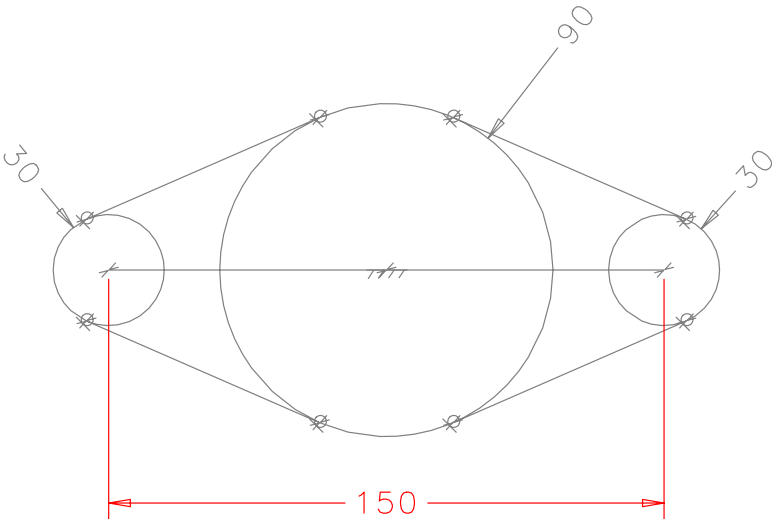


 Turn off *Align* while sketching the tangent lines so that the tangent condition is recognized before alignment with other points.

**Recovery Point**



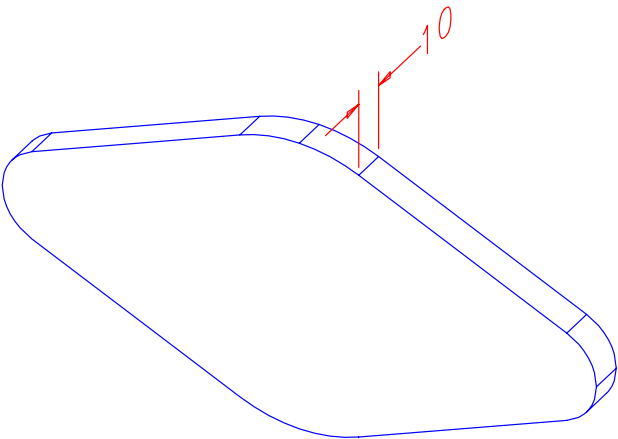
Check to make sure that the wireframe is properly constrained by modifying the length dimension.



Recovery Point



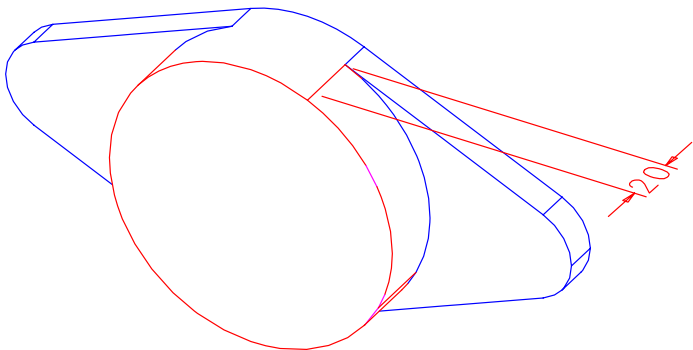
Extrude to the distance shown.



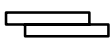
Recovery Point



Sketch in place on the surface shown. Sketch a circle and extrude (join).

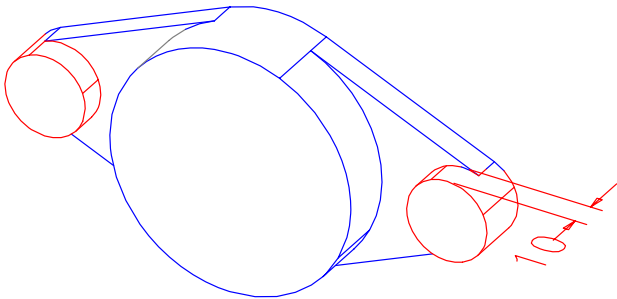


Recovery Point



File  
Save

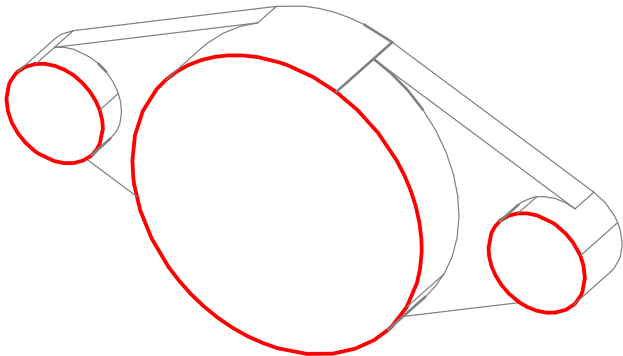
Repeat with two smaller circles.



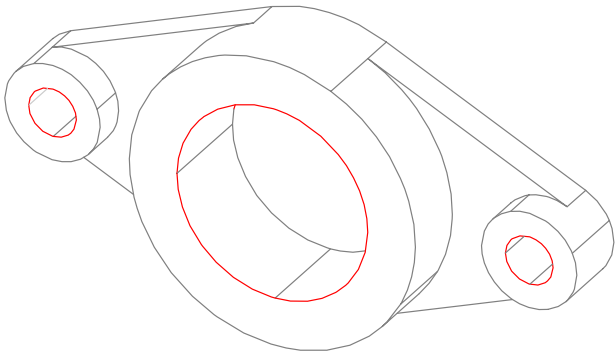
Recovery Point



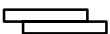
Sketch in place on the three surfaces shown.



Create circles on the three surfaces and cut the holes (extrude–cut).

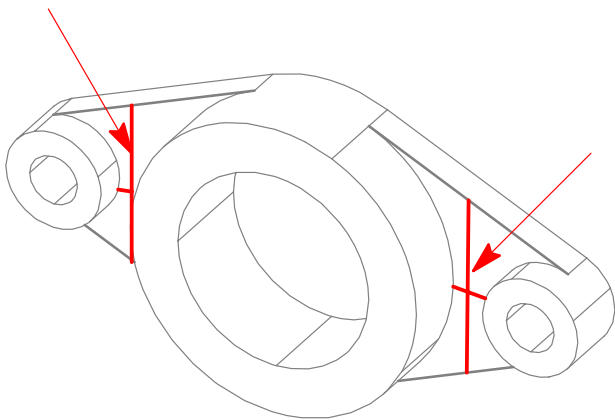


**Recovery Point**



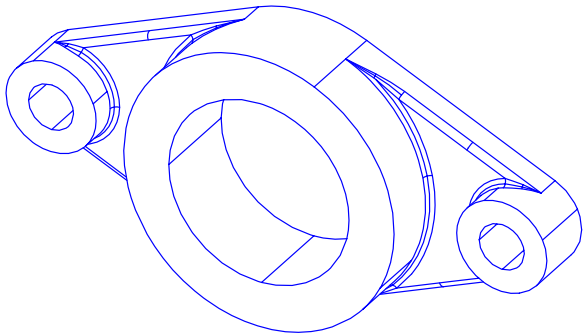
*File*  
*Save*

Fillet the surfaces shown (2mm).

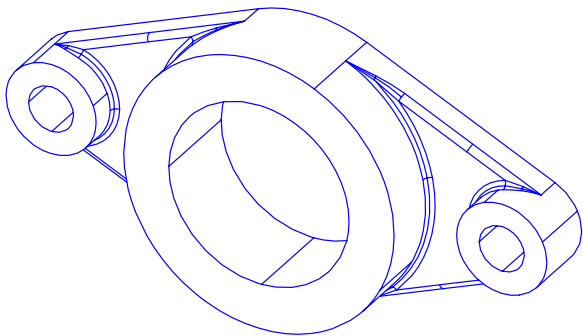


**Hint**

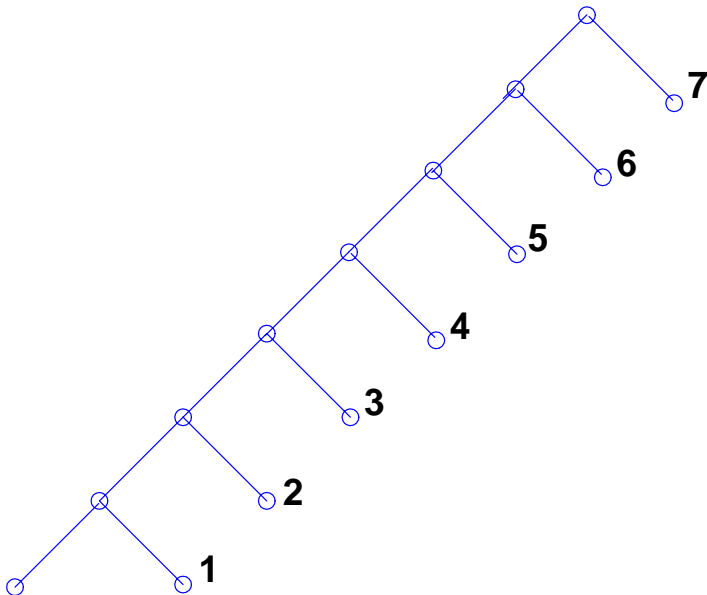
Picking the flat surface will fillet all of its edges.



Finished part.



Check the history tree.



- 1. extrusion
- 2. shaft boss
- 3. mounting hole bosses
- 4. cutout shaft hole
- 5. cutout mounting hole
- 6. cutout mounting hole
- 7. fillet

## Tutorial wrap-up

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You have completed the Creating Basic Machine Parts tutorial.

You can delete or put away the parts. These parts are not used in any other tutorials.